特别说明

此资料来自百度文库（http://wenku.baidu.com/）

您目前所看到的文档是使用的**抱米花百度文库下载器**所生成

此文档原地址来自

http://wenku.baidu.com/view/4c888e29647d27284b735106.html

感谢您的支持

抱米花

http://blog.sina.com.cn/lotusbaob

VISIT…

LANZAROTE
Caliente.COM

Tutorial: Heat and Mass Transfer with the Mixture Model

Purpose

The purpose of this tutorial is to demonstrate the use of mixture model in FLUENT 6.0 to solve a mixture multiphase problem involving heat and mass transfer.

Prerequisites

This tutorial assumes that you are familiar with the FLUENT interface, and have a good understanding of basic setup and solution procedures. This tutorial will not cover the mechanics of using the models; instead, it will focus on the application of the models. If you have not used the mixture models before, the FLUENT Tutorial Guide will provide you with the necessary experience.

Problem Description

The problem to be solved in this tutorial is shown in the following Figure 1.

Initially, the container contains water (the primary phase) at a temperature near the boiling point (372 K). The center portion of the bottom wall of he container is at a temperature higher than the boiling temperature, at 573 K. Because of conduction, the temperature of the fluid near this wall will increase beyond the saturation temperature (373 K). Vapor bubbles will form and rise, due to buoyancy, establishing a pattern similar to a bubble column with vapor escaping at the top and water recirculating in the container.

The default mixture model, as implemented in FLUENT 6.0, does not solve for mass transfer between phases. Therefore User Defined Functions (UDFs) are necessary to model the mass transfer from the liquid to the vapor phase. This is done by incorporating a user defined mass sink term in the vapor continuity equation and an equivalent mass source terms in the liquid continuity equation.

© Fluent Inc. May 16, 2002

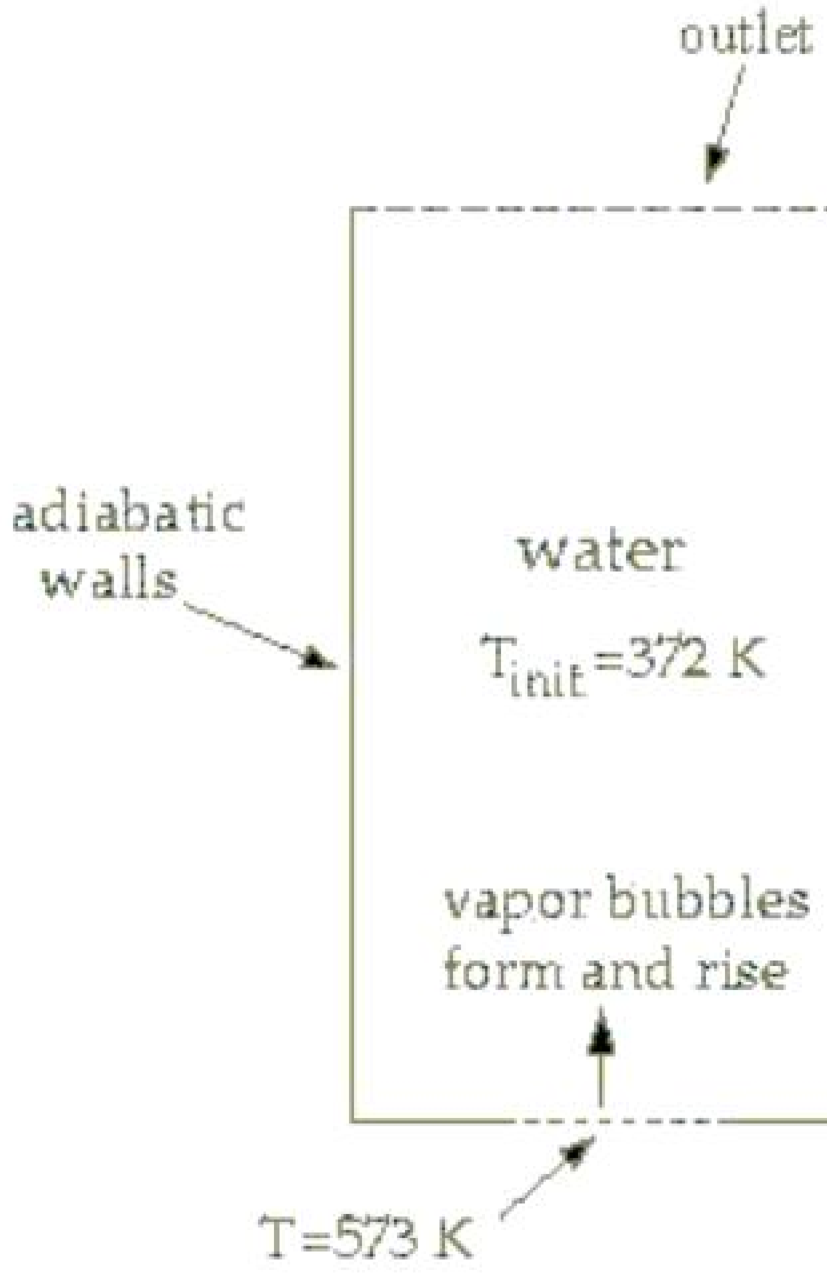


Figure 1: Problem Specification

© Fluent Inc. May 16, 2002

Setup and Solution

Step 1: Grid

1. Read in the grid file boil.msh.

2. Check and display the grid.

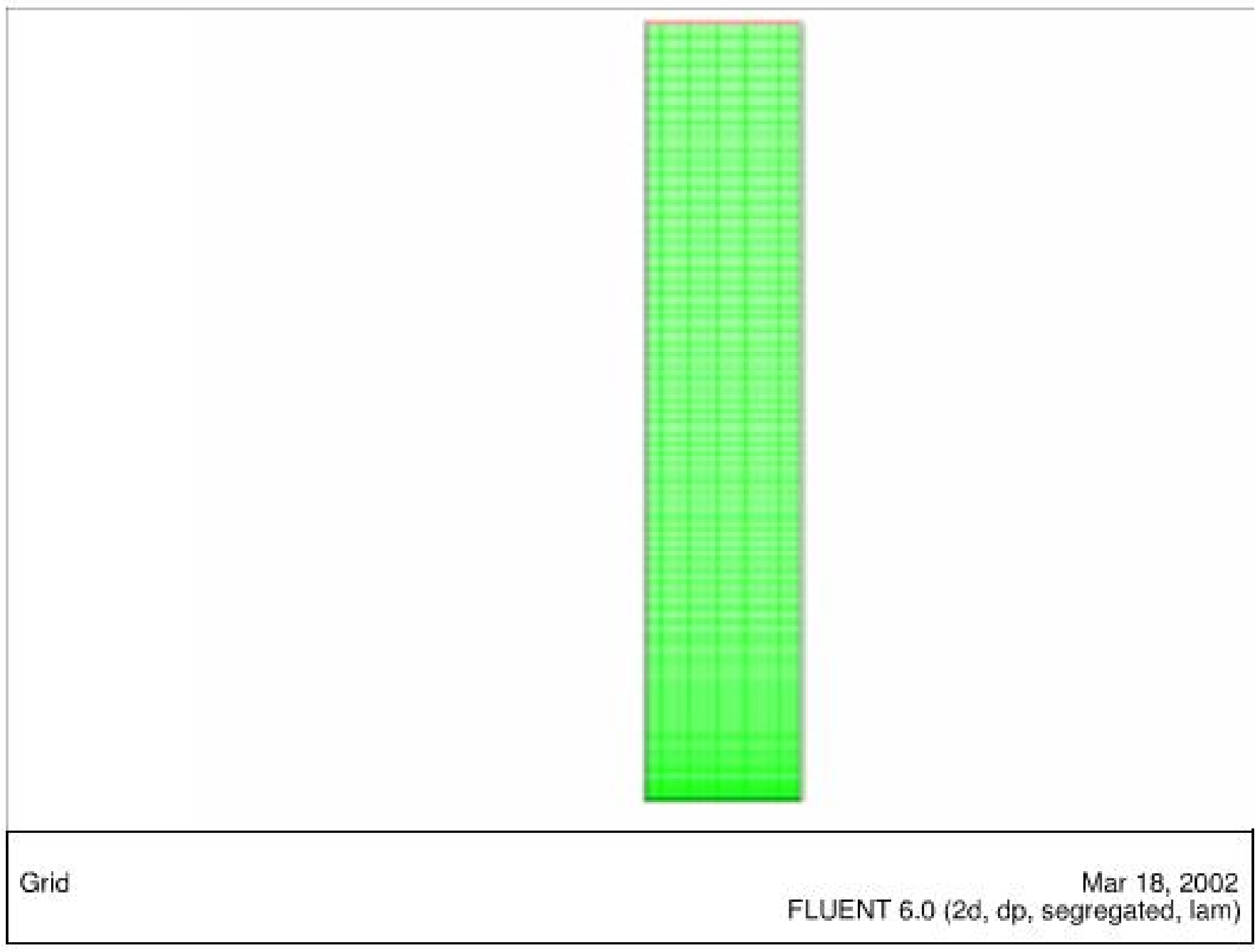


Figure 2: Graphics Display of the Grid

Step 2: Models

1. Define Segregated solver with 2D space and Unsteady time condition.

2. Define the Mixture multiphase model and turn on the Impilicit Body Force.

3. Enable heat transfer by activating the Energy Equation.

Step 3: Materials

1. Modify the properties for air.

   (a) Density=0.5542

   (b) Cp=2014

   (c) Thermal Conductivity=0.0261

   (d) Viscosity=$1.34e^{-05}$

2. Copy water-liquid from the database.

   (a) Change the values for Density and Viscosity to 1000 and 0.0009 respectively. Retain the default values for the other parameters.

3. Define the primary and secondary phases.

   Define ⟶Phases...

   (a) Set the phase material of primary-phase (phase-1) to water-liquid. Rename phase-1 to liquid.

   (b) Similarly set the secondary-phase material to air and rename it vapor. Change the value of Diameter to 0.0002.

Step 4: Operating Conditions

1. Turn on Gravity and under Gravitational Acceleration, enter value of Y as -9.81.

2. Turn on Specified Operating Density, and set the value of Operating Density to that of vapor

   0.5542 ?

Step 5: Boundary Condition

1. Set the boundary conditions for poutlet zone.

   (a) In the Boundary Conditions panel, select mixture for Phase. *The* Pressure Outlet *panel appears.*

   (b) Set the values of Gauge Pressure and Backflow Total Temperature to 0 and 372 respectively.

   (c) For vapor phase, set the value of Backflow Volume Fraction to 0.

   (d) Set the value of Temperature for wall-hot to 573 K.

   (e) Set Heat Flux for all other walls to 0.

   (f) Save the case and data files

Step 6: Compile the Interpreted UDF's

The UDFs are used to specify mass transfer between phases. If you want to use a mass transfer model that is not available in FLUENT*, you can specify it yourself using a user-defined subroutine.*

The mass transfer terms are equal and opposite and are specified as source terms in the volume fraction equations. In addition, there is an energy source term to take into account the latent heat absorbed/released.

For more information on interpreted UDF's, refer to Section 7.2, Interpreted UDFs, *in the UDF Manual.*

© Fluent Inc. May 16, 2002

1. Create a working directory. Save the C functions in your working directory.
2. Start FLUENT from your working directory.
3. Read the case file.
4. Compile the UDF using the Interpreted UDFs panel.
   (a) Enter the name of the C function (source.c) under Source File Name.
   (b) Specify the C preprocessor to be used in the CPP Command Name field.
   (c) Keep the default Stack Size setting of 10000, unless the number of local variables in your function will cause the stack to overflow. In this case, set the Stack Size to a number that is greater than the number of local variables used.
   (d) Select the Use Contributed CPP option if you want to use the C preprocessor that Fluent Inc. has supplied, instead of using your own.
   (e) Click Compile.
   (f) When the compilation is over, click Close to close the panel.

   If you keep the panel open, the Compile *button can be used repeatedly while you are in the process of debugging your function, since you can make changes with an editor in a separate window, and continue to compile until no errors are reported.*

Step 7: Assign the UDF's

Define the macro's for liquid, vapor and mixtures.

1. Select fluid for Zone and liquid for Phase. Click Set....

   The Fluid *panel opens.*
2. Turn on Source Terms and set the value of Mass to udf liq_src.
3. Similarly, in the Fluid panel for vapor, set the value of Mass to udf vap_src and for mixture, set the value of Energy to udf enrg_src.

Step 8: Solution

1. Set the parameters that control the solution.
   (a) Retain the default selected equations (all of them).
   (b) Under Under-Relaxation Factors, set the values of Pressure to `0.5`, Momentum to `0.2` and `Volume Fraction` to `0.4`.
   (c) Under Discretization, set Pressure interpolation scheme to Body Force Weighted and change Momentum and Volume Fraction to Second Order Upwind.
   (d) Retain the values for other parameters.
2. Initialize the flow field Temperature to 372 K.
3. Adapt the boundary region next to wall-hot.

© Fluent Inc. May 16, 2002

(a) Under Boundary Zones, deselect all the zones and then select wall-hot.

(b) Set Number of Cells to 1 and click on Mark to mark the cells for refinement.

4. Patch a temperature slightly higher than the saturation temperature of 373 defined in the UDF.

(a) Select Temperature for Variable and boundary-r0 for Registers To Patch.

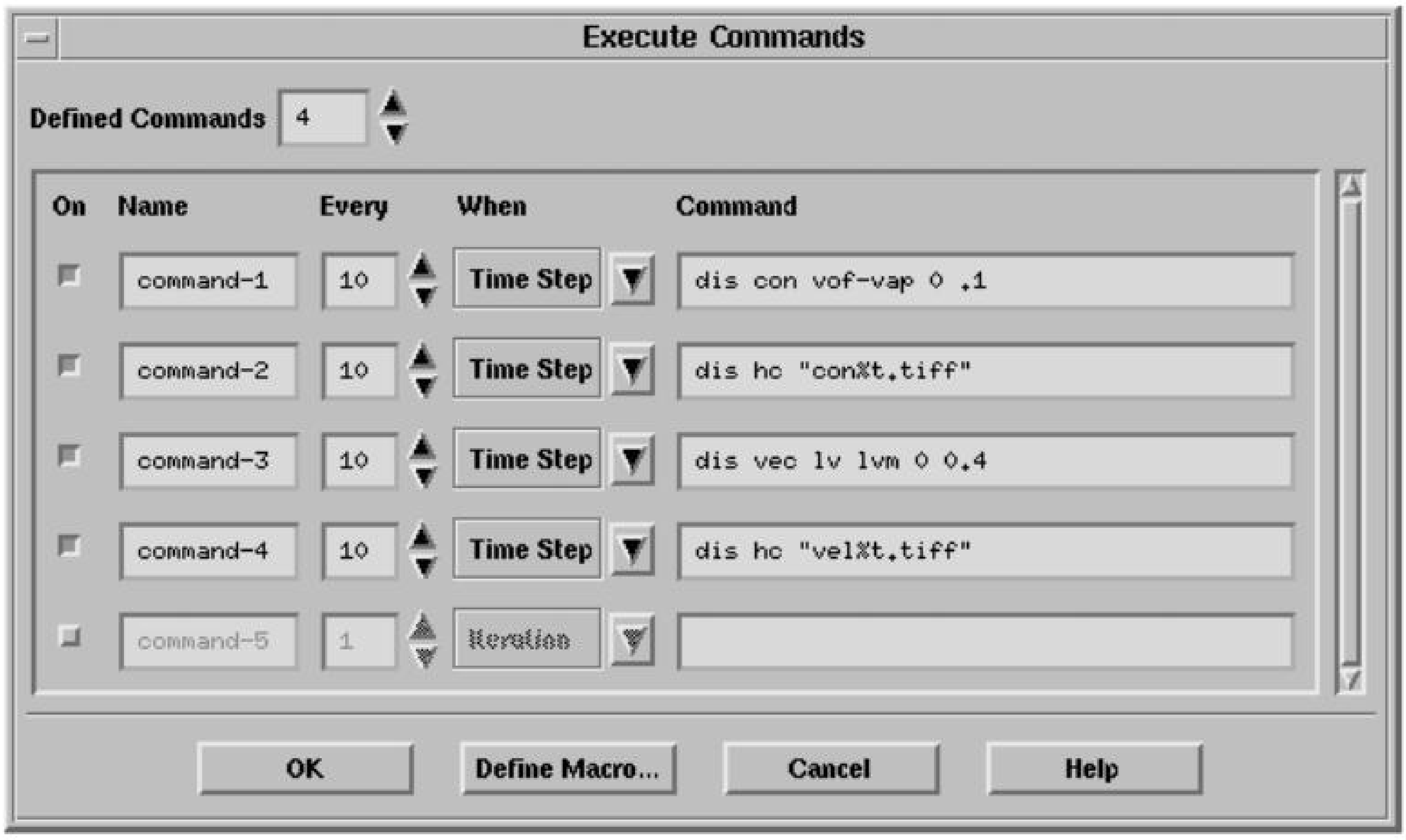
(b) Set the Value to 373.15 and click on Patch.

5. Set commands for animations

Solve ⟶Execute Commands...

If required, set up commands to write out tiff files for animation.

The Execute Commands panel is displayed. Define the commands as shown in the following panel.



6. Set the Time Step Size to 0.01 and Number of Time Steps to 1000 and click Iterate.

© Fluent Inc. May 16, 2002

Step 9: Postprocessing

1. Display filled contours of liquid velocity magnitude.

   (a) Select Velocity... and liquid Velocity Magnitude in the Contours Of drop-down list and click Display.

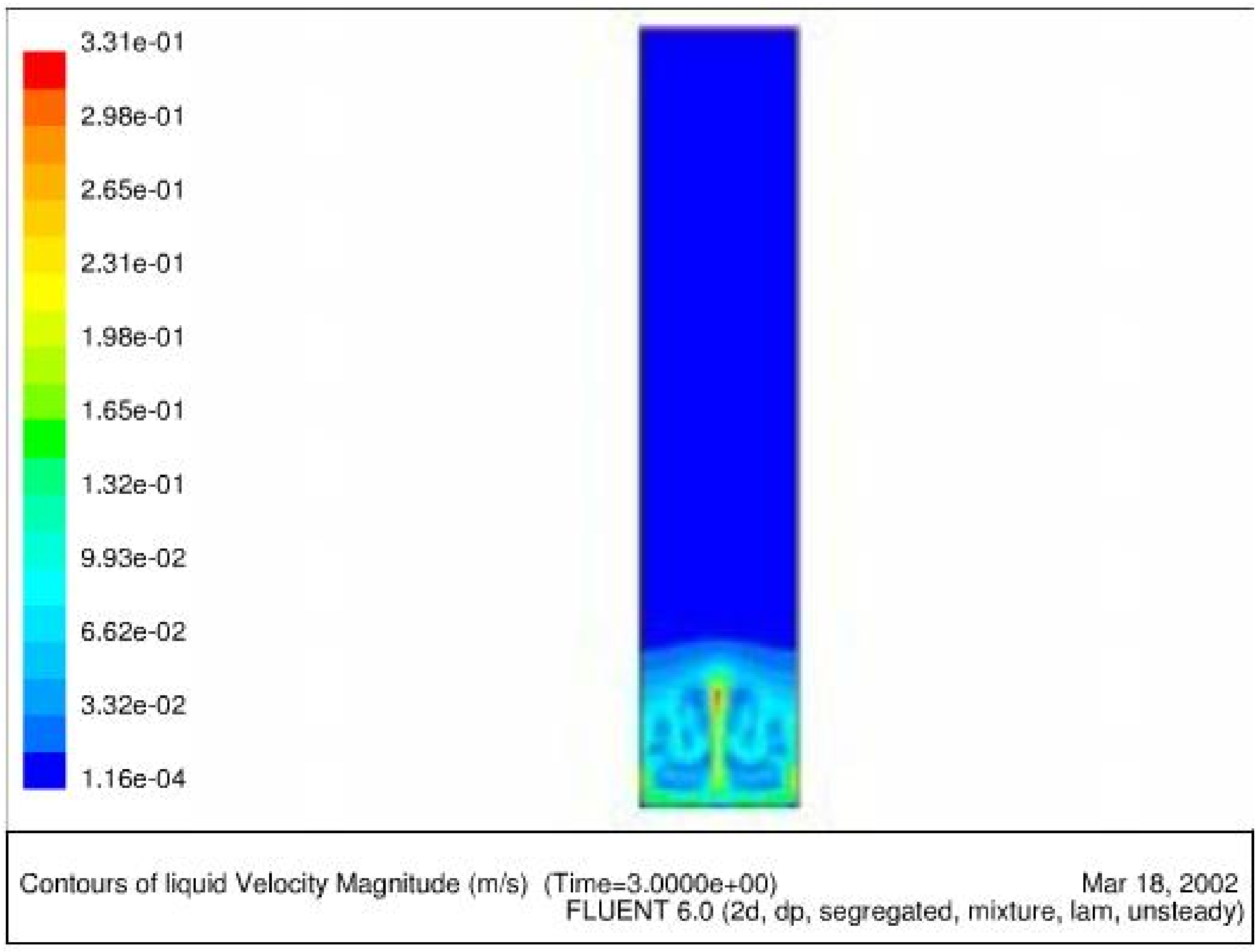


Figure 3: Contours of liquid Velocity Magnitude

2. Display filled contours of vapor volume fraction.

   (a) Select Phases... and Volume fraction of vapor in the Contours Of drop-down list and click Display.

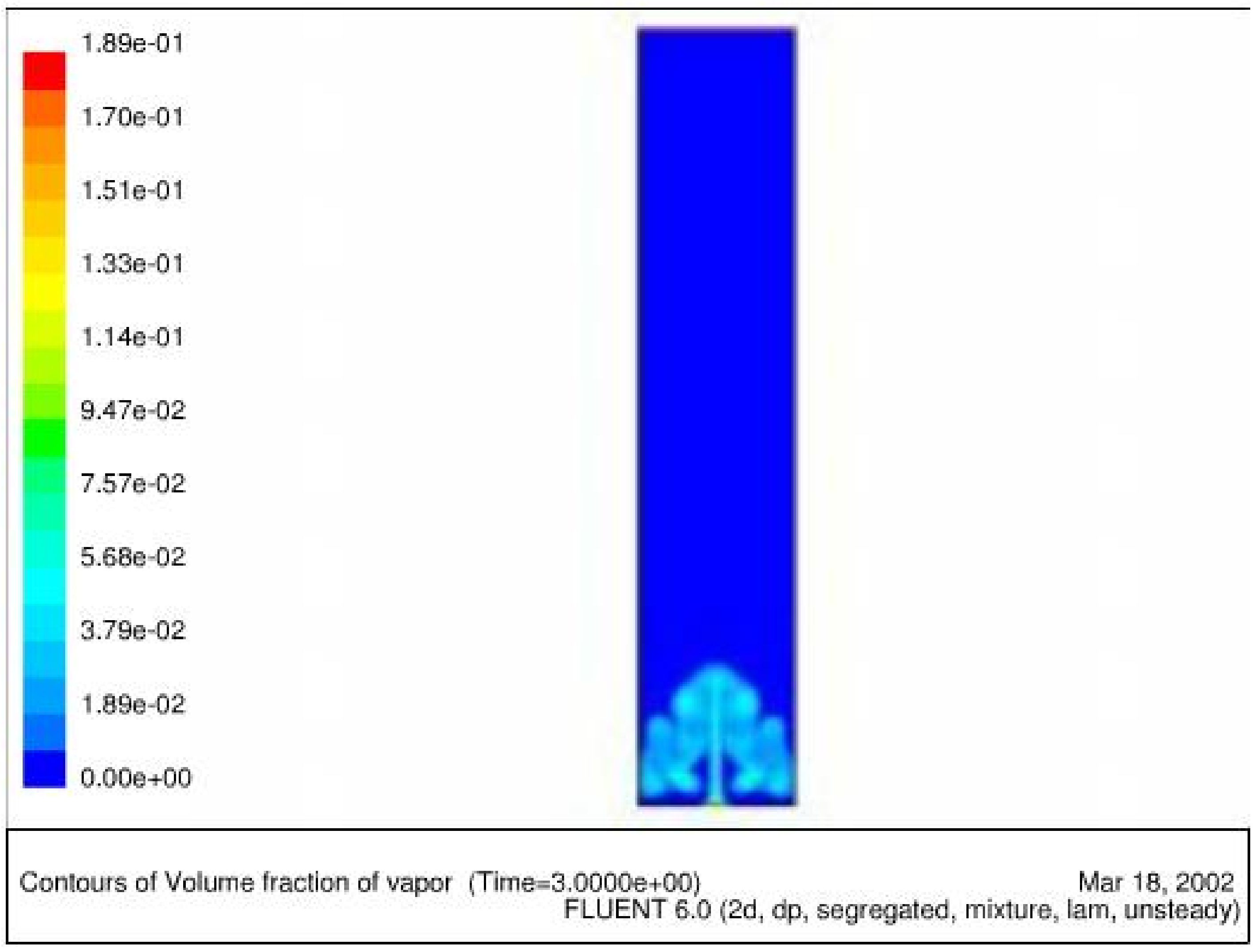


Figure 4: Contours of Volume fraction of vapor

© Fluent Inc. May 16, 2002

3. Display filled contours of static pressure.

   (a) Select Pressure... and Static Pressure in the Contours Of drop-down list and click Display.

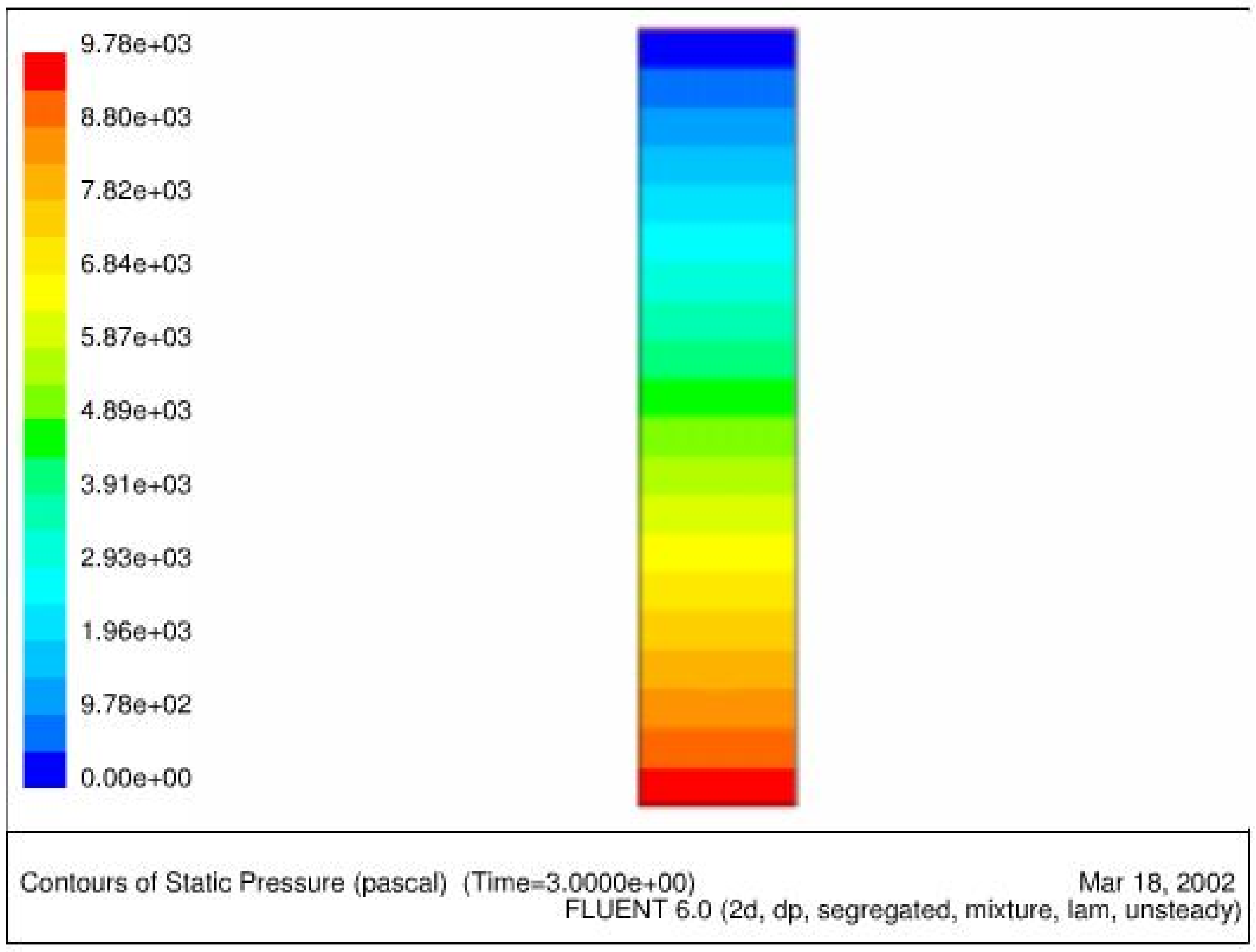


Figure 5: Contours of Static Pressure

4. Display filled contours of static temperature.

   (a) Select Temperature... and Static Temperature in the Contours Of drop-down list and click Display.

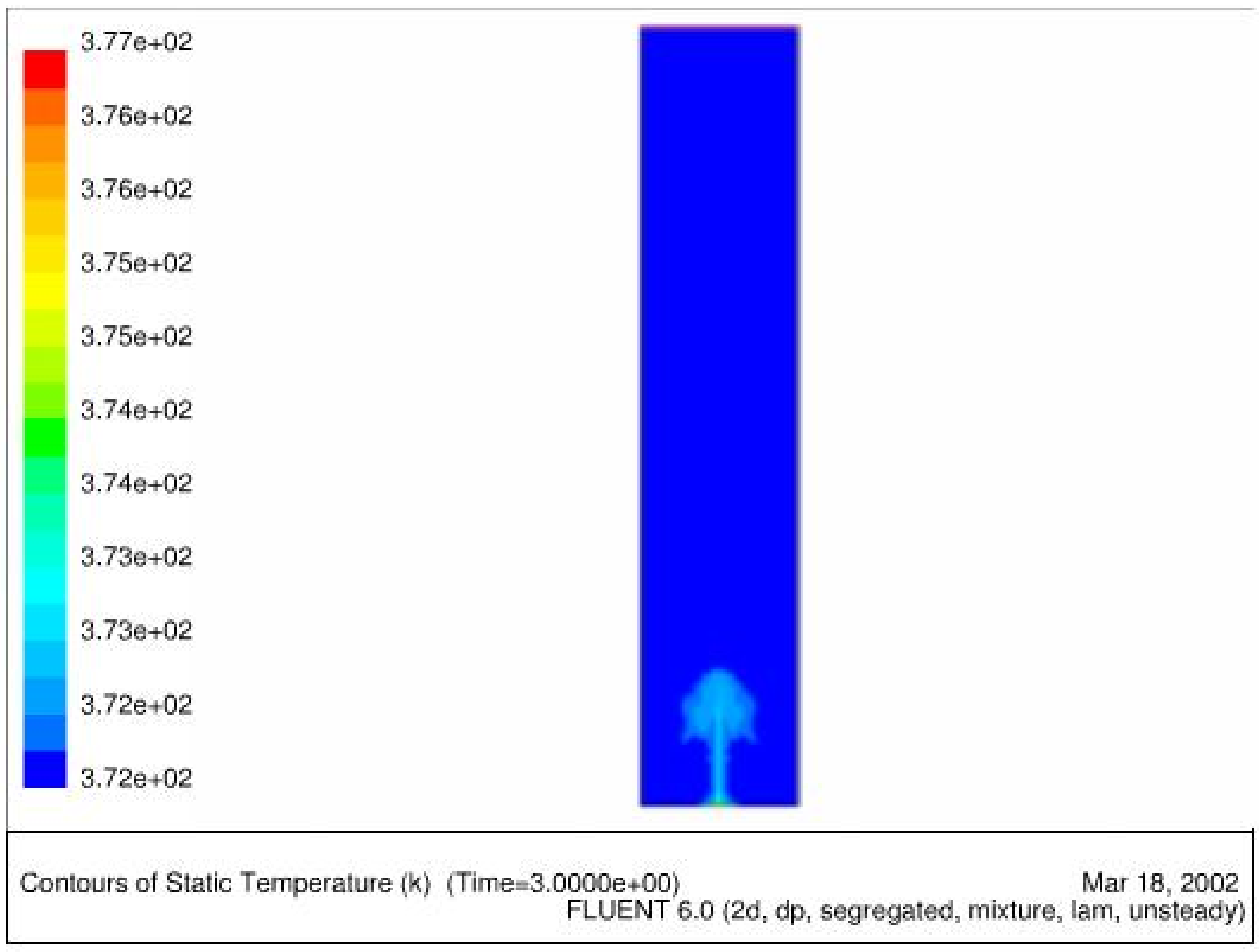


Figure 6: Contours of Static Temperature

© Fluent Inc. May 16, 2002

Results

When you use interpreted UDFs, the name and contents of the `source.c` will be stored in the case file, and when you read the case in a later session, the routine(s) will be compiled during the reading process.

Summary

Application of the ASMM to solve a mixture multiphase problem involving heat and mass transfer has been demonstrated in this tutorial. Also, UDF's have been used to enhance the standard features of FLUENT.

© Fluent Inc. May 16, 2002